



INVESTIGATION OF HYDRAULIC PERFORMANCE OF A FLAP TYPE CHECK VALVE USING CFD AND EXPERIMENTAL TECHNIQUE

Nagaraj Sitaram

MVJ College of Engineering, Bangalore

Abhijeeth Nagaraj, Adarsha H, Kaushik V Prasad and V Ravinarayan

Department of Mechanical Engineering,

School of Engineering and Technology, Jain University, Bangalore

ABSTRACT

An attempt has been made to simulate flow through an 820-mm diameter slanting disc check valve by using CFD. The simulations are carried out for various disc angles with different flow velocities. The pressure drops (ΔP) and the average flow velocity (V) are used to determine the pressure loss coefficient (K_L) for each case, and the obtained analytical values of " K_L " have been compared with the experimental results. The computational analysis has been carried out at five different disc angles of check valve i.e. for 54° (fully opened), 46° , 34° , 24° , 10° . The pressure drops (ΔP) and the pressure loss coefficient (K_L) are computed for all the five cases and compared with experimental values. The obtained numerical results are in close agreement with experimental values, for higher disc angle openings (Greater than 10°). The improved design is modeled and tested in the laboratory which has shown an improvement of 10% of pressure loss in the valve.

Key words: Check valve, CFD, Loss coefficient, pressure drop, Disc angle, valve size, valve seat, experiment, Turbulence model.

Cite this Article: Nagaraj Sitaram, Abhijeeth Nagaraj, Adarsha H, Kaushik V Prasad and V Ravinarayan, Investigation of Hydraulic Performance of A Flap Type Check Valve Using CFD and Experimental Technique, International Journal of Mechanical Engineering and Technology, 10(1), 2019, pp. 409–413.

<http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=10&IType=1>

1. INTRODUCTION

Check valves are operated entirely by the reaction of the line fluid and are installed when flow is pumping main failures of liquids in pipeline. The valves can be classified and analyzed into various categories. The check valves are commonly based on either the swing design or lift design^[1]. The swing check valves utilizes a flap or disc of same diameter as the bore of the pipe which hangers down and is allowed to swing in the flow of fluid. Any reverse in the flow will cause it to slam against the seat and stop the flow going back down the pipe.

The improvements in the valve requires deep insights into the valve flow and their behavior with various angles of opening and during its operation. The insights are key in improving the valve performance by understanding their complex flows^{[2],[3]}. Researchers have concentrated on the dynamic behavior of the flap disc and showcases a use of coefficients to understand the valve performance^[4]. The experimental work has been used to generate an analytical model using empirical results to determine the valve operation parameters^{[5][6]}. But the work defines no experimental relationship to the net pressure drop with respect to the flow pressure.

The work suggests the use of experiment to validate the CFD model and utilize for prediction under operation conditions. The method also tries to explain the conditions the expected errors during simulations.

2. EXPERIMENTAL SETUP

The test valve was installed with 600 x 820 mm expander and 820 x 600 mm reducer with 10m upstream, 6.4m downstream pipe length as shown in Figure 1. The interfacing portion was removed by chipping/grinding and valve disc was made to open fully (54°). The 24" (600mm) pipeline was flooded and entrapped air was cleared through air valves. Disc angle of the check valve under test was measured using angle sensor having a readability of better than 0.1°. Differential pressure across the valve was measured using a floating piston manometer with readability of 0.2millibar. The flow rate through the valve is measured using a 20-Ton weighing machine /diverter (gravimetric) system. Time of collection of water is taken through high a precision counter. Line pressure is measured using a precision pressure gauge.

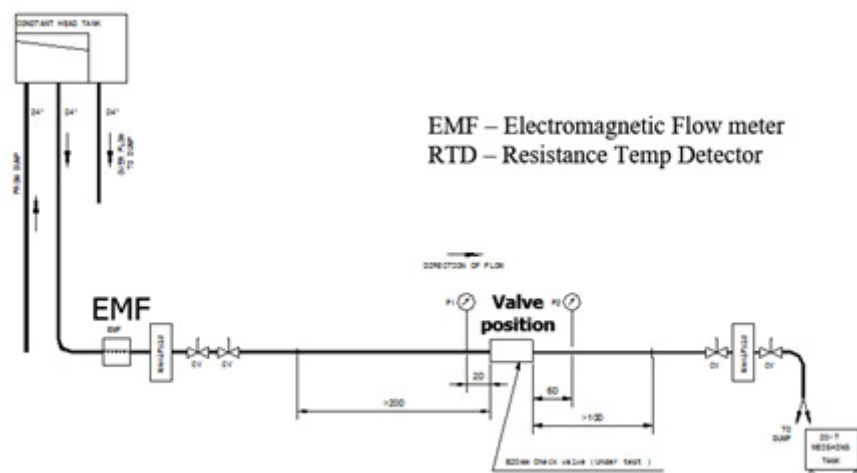


Figure 1 Experimental setup

3. EXPERIMENTAL TEST

The experimental method involves in determining the disc angle of the valve corresponding to this velocity. The water was collected for a specified duration. The collection time,

differential pressure, weight (initial and final readings) of collected water and density of water (corresponding to water temperature) were noted.

The pressure loss due to the valve is determined using a valve loss coefficient (K_L). The experiment test has been conducted to determine the K_L of the given valve. The experiment involved in determining the pressure drop against the flow rate. The equation of calculation for K_L is given as per equation 1

$$K_L = \frac{\Delta p}{\left(\frac{1}{2} \rho_f V^2\right)} \quad (1)$$

Where,

Δp = Pressure loss across the Valve in N/m^2

ρ_f = Density of the fluid kg/m^3

V = Velocity of flow in m/s

Table-1 shows the valve loss coefficient (K_L) for various Disc opening of the valve. The maximum flow velocity obtained for static performance was 3.05 m/sec and corresponding disc angle of check valve was 54° from closed position. The flow velocity was also computed from the flow rate for various disc opening of the check valve. This is also shown in Table1.

Table 1 Loss Coefficient from experiment

SI No.	Disc angles ' θ ' (in degrees)	Flow velocity 'V'(m/s)	ΔP (Pa)	Loss coefficient (K_L) Experimental
1	10	0.27	2170.60	59.55
2	25	0.95	2301.38	5.100
3	34	1.50	2486.25	2.210
4	46	2.37	2763.51	0.984
5	54 (FullyOpened)	3.05	2972.15	0.639

4. FLOW SIMULATION

The computational fluid dynamics (CFD) as the potential to significantly reduce the time and expense in determining flow behavior of valves. This paper explores the ability of CFD to match the loss coefficient ' K_L ' of a check valve with experimentally determined values for various angles of openings.

4.1. Modeling Technique

The computation is carried by using a CFD code based on the finite volume method, which as a separate pre-processing, post processing modules. The Reynolds number based on the diameter ($D = 820mm$) and mean velocity of flow (with water as a flow media) ranged from 22, 1400 to 2.501×10^6 the operating Reynolds number is in the fully turbulent flow region.

4.2. Geometry

The pre-processor is used for geometric modeling and meshing. In the present problem the geometry of the check valve is modeled in 3-dimensional and consist of 497,688 elements (Tetrahedral mesh using Hex cooper schemes). The number of grid elements was increased to check the sensitivity of results on number of grid elements but results were not found to vary significantly (particularly the values of ' K_L '). Figure 3 shows a meshed geometry of the check valve. The following are the assumptions which are made for simulating flow through the check valve.

- The dimensions of the valve had been taken approximately with respect to the disc dimensions
- The fluid density and viscosity are constant.
- The fluid is Newtonian, incompressible, and homogeneous.
- The turbulent flow can be modelled by using k-epsilon (k- ϵ) and RNG k- ϵ model.

The valve geometry is complex and care taken during meshing to avoid cell jump and negative volumes. Low pressures are expected to occur in the separated flow region behind the valve body as shown in figure 3. To obtain better resolution fluid cell densities are increased in the vicinity of the separated flow region. A total length of 18m (Upstream length =10m, Down-stream length = 6.4m, valve body and flanges =1.6m) is taken for the flow modeling of check valve.

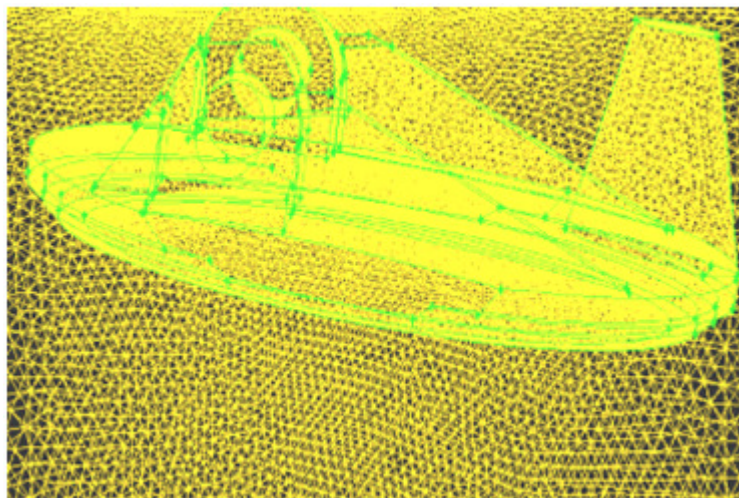


Figure 3 Meshed model of the valve

4.3. Solver Conditions

A three-dimensional segregated solver with implicit steady scheme is used. The 'SIMPLE' algorithm is used to couple pressure velocity corrections while 'standard' scheme is used to calculate to pressure of the cell faces. The maximum inlet turbulent intensities (based on Reynolds number) are estimated about 4% with hydraulic diameter as 820 mm. The gravity effects are considered for the present simulation. The turbulence models standard k- ϵ and RNG k- ϵ are used with standard well correction procedure. A converged solution is obtained based on the mass flux residuals (0.1% variation).

The following boundary conditions applied to the present problem

- Velocity inlet(m/s)
- Outflow
- No-slip boundary conditions at the walls.

The computational analysis as been done at five different disc angle of the check valve i.e. for 54° (fully opened), 46°, 34°, 24°, and 10°. The pressure drop (Δp) and the pressure loss coefficient (K_L) are computed for all the five cases.

5. RESULTS AND DISCUSSIONS

The computational analysis has been done at five different disc angles of the check valve i.e. for 54° (fully opened), 46°, 34°, 24°, and 10°. The change in pressure (Δp) and the pressure loss certificate (K_L) are computed for all the five cases.

Figure 4 highlights the values of obtained loss coefficient with respect to disc angles and the experimental values. The computed value of loss coefficient is compared with the experimental results, it has been observed that the obtained numerical results are in close accordance for higher disc angle openings (Greater than 10°). The deviation is higher for lower disc angle openings; this may be due to the assumption in the valve dimensions (which is already mentioned in the list of assumption).

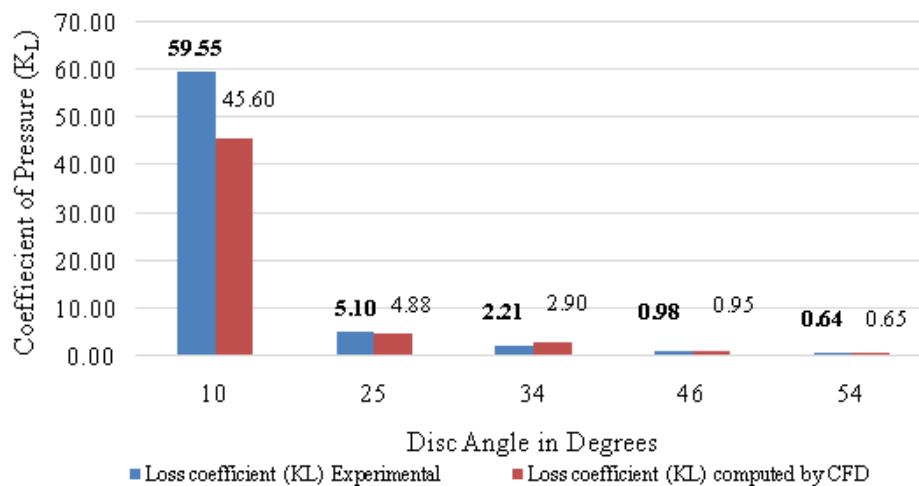


Figure 4 Loss Coefficient comparison of experimental and simulation methods

The flow distortions behind the valve body were also seen in detailed experimental study (through pressure measurements). The main findings after the experiment on check valve are:

- It has been observed that the Grid Density and mesh quality improvements are the two important factors for obtaining better results in check valve simulation
- The computed value of the loss coefficient (K_L) agree well for higher angles of disc openings but the values found to have less variation with lower angles of openings (For less than 15° of valve openings)

ACKNOWLEDGEMENTS

Author would like to acknowledge the support and co-operation provided by the staff of Water Flow Laboratory of FCRI, Palakkad (Kerala)

REFERENCES

- [1] Skousen, P. L., *Valve Handbook*, McGraw-Hill, New York, 1998
- [2] Versteeg, H. K., and Malalasekera, W., *An Introduction to Computational Fluid Dynamics. The Finite Volume Method* Longman, England, 1995
- [3] Chung, T. J., *Computational Fluid Dynamics*, Cambridge University Press, Cambridge, 2002
- [4] Ellis, J., and Mualla, W., Numerical modeling of reflux valve closure. *J. Pressure Vessel Technol.* 108(1), 1990, 92–9
- [5] Li, G. and Liou, J.C., Swing check valve characterization and modeling during transients. *Journal of fluids engineering*, 125(6), 2003, pp.1043-1050.
- [6] Turesson, Martin, *Dynamic simulation of check valve using CFD and evaluation of check valve model in RELAP5*, M.Sc. Thesis, Chalmers University of Technology, 2011